IV International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2011
M. Papadrakakis, E. Oñate and B. Schrefler (Eds)

SIMULATION OF RESIDUAL STRESSES IN AN INDUCTION HARDENED ROLL

CLEMENS GROTH[*], LUDWIG HELLENTHAL[†]

[*] CADFEM GmbH
Hannover office
Pelikanstraße 13, 30177 Hannover, Germany
e-mail: cgroth@cadfem.de, www.cadfem.de

[†] WALZEN IRLE GmbH
Hüttenweg 5, 57250 Netphen-Deuz, Germany
email: hellenthal@walzenirle.com, www.walzenirle.com

Key words: Induction, microstructure, transformation induced plasticity, heat treatment, induction hardening, weld simulation

Abstract. Hardening processes are directed towards improving the metallurgical conditions of steel parts. The thermal process is directly coupled with mechanical consequences due to the thermal elongation and plastic mechanical behaviour. The metallurgical transformation of the steel results in material data changes that further complicate the simulation of the process. A simulation application is presented where the heat is induced by an electrical inductor. The part under consideration is a calender roll for a paper production site. The inductor encloses the roll and travels along the length. A coolant flow is applied directly behind the heated zone. Due to the induction, electrical and magnetical effects are significant for the heat source distribution and have to be included in the simulation. An overview over the coupling effects is presented.The simulation steps are directly and sequentially coupled. The result of the simulation is the temperature distribution over time and space and the residual stress distribution. The simulation results are compared with experimental results of residual stresses. The coincidence is satisfactory.

1 INTRODUCTION

The structure under consideration is a roll for the plastics industry. Rolls like the one mentioned here are used pairwise to produce very thin plastic sheets. Due to the intended use of the roll the surface must be manufactured regarding extreme accuracy requirements. This applies to the surface roughness as well as to other dimensions like excentricity and cylindricity under cold and hot (200°C) conditions. Additionally, the operating loads include a significant bending moment and heat expansion, which leads to tension stresses at the roll surface. A compression residual stress can help to neutralize these operating stresses and thus reduce the possibility of crack initiations.

The size and cost of the roll is significant. The simulation is very advantageous to investigate the heat treatment process and parameter variations beforehand.

2 THE STRUCTURE

The roll outer diameter is 900 mm, the axial length exceeds 2000 mm. After the forging and annealing pocesses the roll outer surfaces are pre machined closly to final domensions. The roll as it is used for the hardening process can be regarded as being massive. The final construction has internal drill holes and openings.

The material is steel comparable to 1.6959. It is a ferritic steel that shows microstructural changes when heated above temperatures of 800°C.

3 THE HARDENING PROCESS

The objective of the hardening process is to achieve a hardness level at the roll surface which is needed to fulfill adequate life cycle and wear requirements. In addition to this it is intended to achieve a certain residual stress level to neutralize the operating stresses.

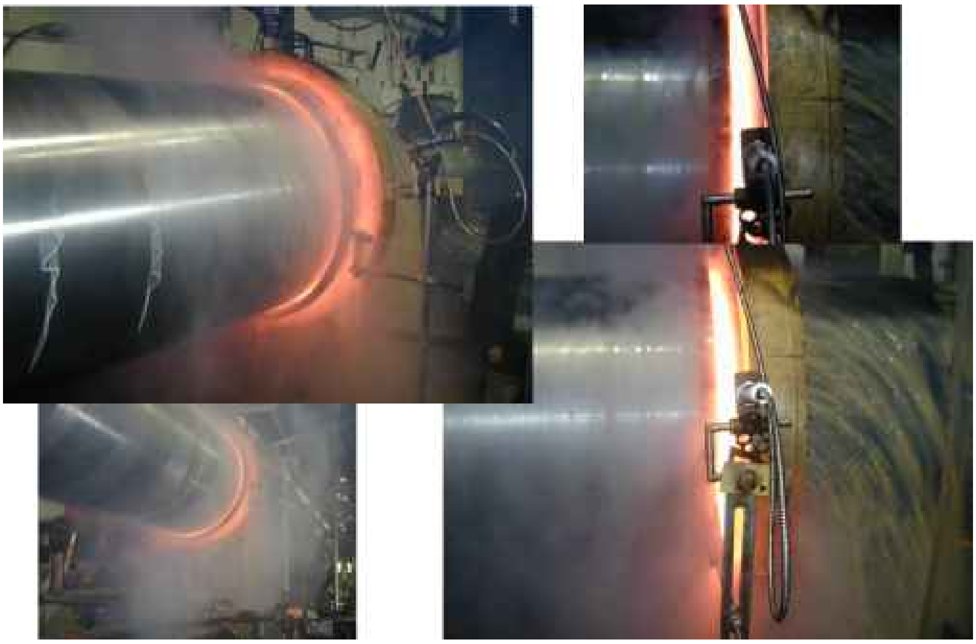

Fig.1 The roll during the hardening process

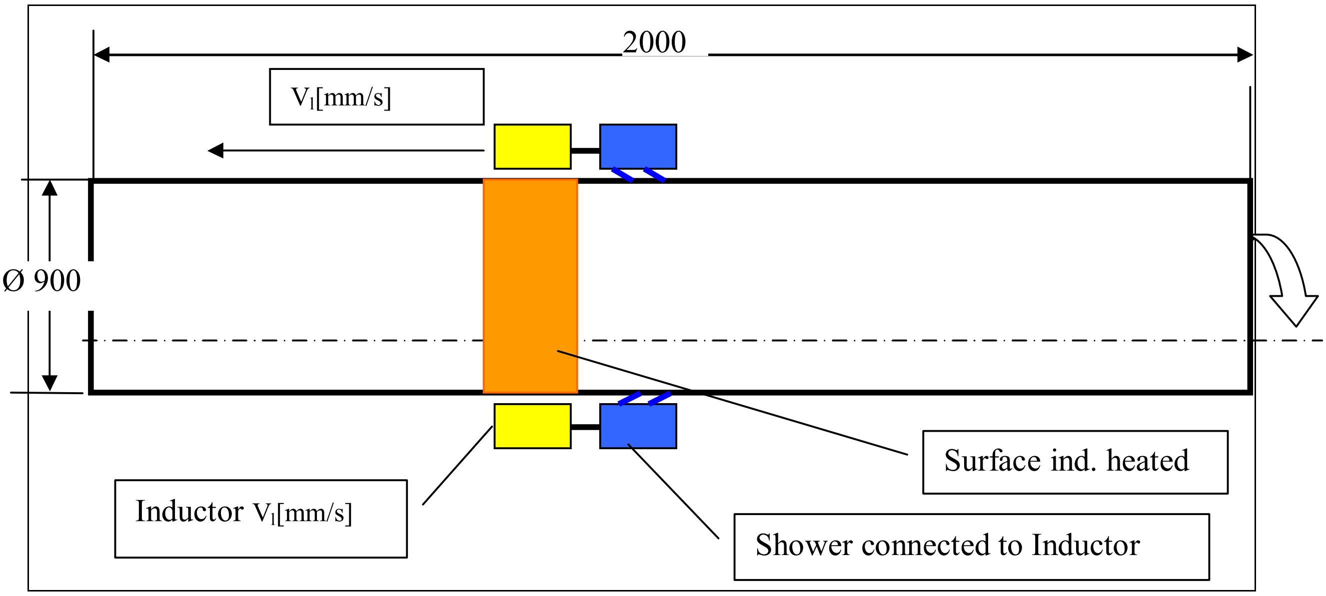

Fig.2 The roll during the hardening process

This hardening process is to be simulated. Economic considerations require the number of variations to be minimized as far as possible. The production of a roll is extremely cost and time consuming. Any variations of process parameters have to be simulated in advance and the results thoroughly evaluated.

The hardening process as shown in Fig. 1 and 2 was performed with the roll as described in chapter 2. The simulation task was to idealize this hardening process and to compare the experimental and simulation results. This comparison was used to qualify the simulation as a decision basis for future variations of the process.

The hardening is done by heating the roll material by means of an induction ring surrounding the roll. An alternating electric current flows through the induction ring and induces a magnetic field around the ring. The induction ring is a hollow rectangular copper section. The magnetic field induces electric currents in the surface region of the roll material. The current losses result in a heat-up. With the appropriate feed rate V_l of the axial movement of the ring the maximal temperature at the roll surface reaches values of close to 1000°C. The inductor ring is slowly moved axially along the roll axis. Following the inductor ring is a water jet channel which sprays a watery coolant on the roll surface. During this process the roll is rotated slowly so that all circumferential sections are treated identically. Surface measurements are used to adjust the process parameters so that the surface temperature is close to 1000°C.

4 PHYSICAL INTERACTIONS

An overview of the physical interactions is given in Fig. 3. The alternating electric current results in an alternating magnetic field (Magnetisches Feld) around the inductor ring. This magnetic field induces electric currents in the metallic roll material (Elektrisches Feld). This electric-magnetic field interaction requires the surrounding air and far field regions to be included in the model.

The thermal field (Temperatur-Feld) is coupled bidirectional. The electric-to-thermal coupling includes the current energy losses in the roll material which act as thermal heat source terms. The reverse thermal-to-electric coupling includes the influences of the temperatures on the material data. The microstructural behavior (Gefügeumwandlung, micro structure transformation) influences all of these field effects. The steel used for the roll is a ferritic steel which changes to austenitic material above a certain boundary temperature and shows a reverse change when a certain boundary temperature (usually lower than the first mentioned boundary temperature) is passed. The material behavior is described by CCT diagram. These material changes influence the material data significantly, especially the magnetic behavior is influenced extremely.

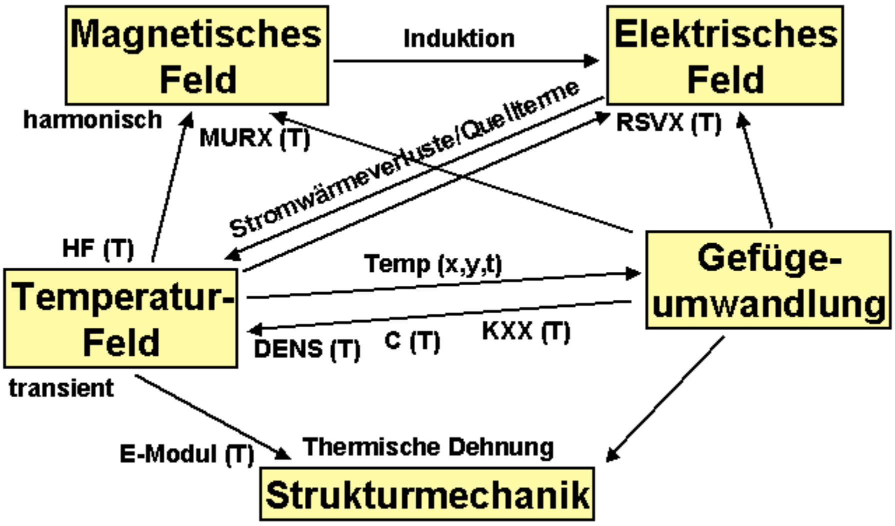

Fig.3 Physical interactions

The mechanical simulation (Strukturmechanik) is coupled uni-directional. The mechanical material values depend on the thermal field and on the microstructural changes, for example the module of elasticity and also the thermal expansion. The thermal expansion is provided as dilatogram graphs, see Fig.6. The graphs are common means to measure the microstructural changes and to document the material behavior.

5 ELECTRIC-MAGNETIC-THERMAL SIMULATION

For the electric-magnetic field a harmonic simulation is used to idealize the alternating current source. The model which was used is shown in Fig. 3. This figure shows the inductor in the centered position along the roll axis. The mesh is axisymmetric and consists of the roll material, the inductor ring cross section and the surrounding air region. Infinite elements are spread over the outer air region boundary to care for the far field effect. To simulate the movement of the inductor ring along the axis this model is used for a sequence of configurations where the inductor plus the surrounding air are moved axially and coupled to the appropriate wall surface mesh nodes. This sequence is repeated for several positions and thus provides a sequence of field distributions for discrete time points. Parallel to this electromagnetic field analysis a transient thermal field is simulated where the electric energy losses are used as heat source terms. Both analyses exchange the heat source terms and the temperature values (for material data adjustment) sequentially resulting in a kind of "explicit" coupling.

The microstructural changes were taken into account by means of an APDL (ANSYS parametric design language) algorithm that represents the CCT diagram information. Temperatures, temperature gradients and bounding temperature limits are considered.

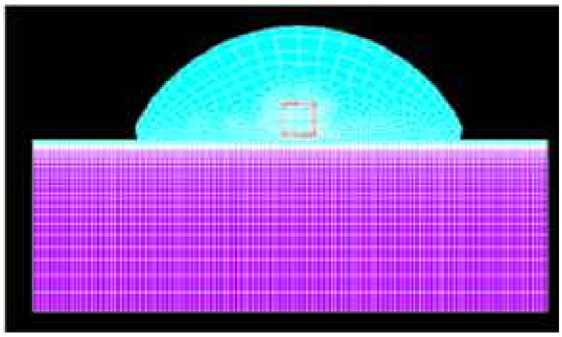

Fig. 3: Model configuration and FEM mesh

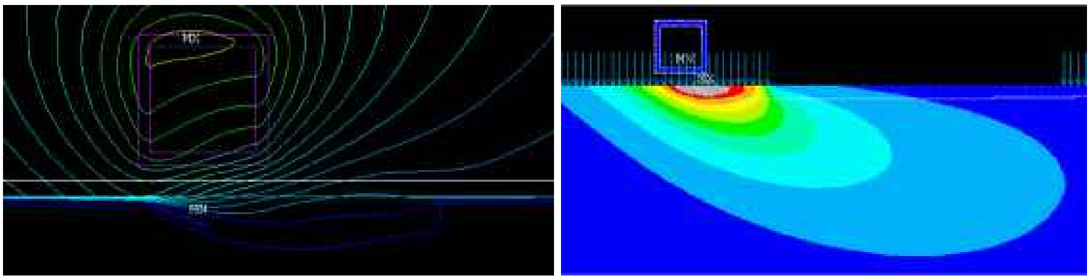

Fig. 4: Magnetic field and temperature distribution at halfway inductor position

6 THE VIRTUAL WELD SIMULATION TOOL (VWS)

The VWS virtual weld simulation tool is a software add-on produced by CADFEM GmbH and partners, funded by BMBF. The tool uses the ANSYS core product as basis and benifits from the numerous capabilities of this general purpose program. The VWS weld simulation tool provides the special features that are necessary to simulate welding processes of any kind. First applications focused on tailored blanks and laser welds. The features of the VWS tool also cover the special requirements of applications like welds with other heat sources (electron beam, friction stir), welds for shipbuilding, commercial vehicles, airplane panels, or heat treatments.

One of the VWS tool features which was used for the induction hardening of the roll was the STAAZ method. This method uses material data representing the thermal expansion of the material with microstructural changes. Common methods to quantify the thermal expansion are dilatograms showing the elongation of a specimen during a heat-up and cool-down cycle. Usually these dilatogram results are used to design the CCT diagram of the material. The STAAZ method requires a set of dilatograms to be provided covering different cases of value tripel with

- maximum temperature during the process,
- austenitization time (time span in the austenitic regime) and
- cooling gradient (time between 800°C and 500°C, t8/5).

The VWS tool uses the dilatogram information and interpolates the thermal expansion in each element according to the local value tripel as extracted from the thermal analysis. The advantage of the STAAZ method is that the primary experimental results are used directly without intermediate steps like designing CCT diagram, extracting analytical function parameters or interpolating in the CCT diagram or other analysis steps. The disadvantage of the STAAZ method is that no information about material fractions like austenite, bainite, martensite or others can be achieved.

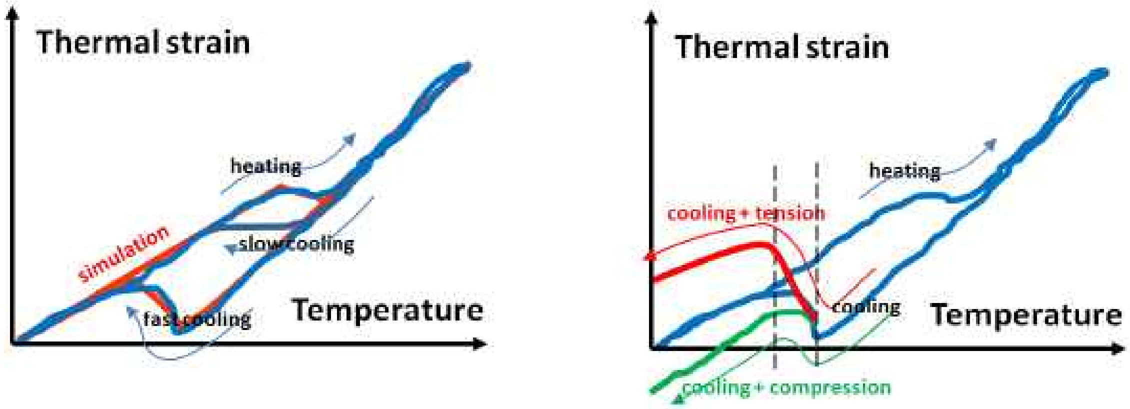

Fig. 5: Typical dilatogram (left: cooling rate, right: TRIP transformation induced plasticity)

Fig. 5 (left) shows a dilatogram result with a variation of cooling rates (fast cooling, slow cooling). When the experiment is done without mechanical stresses (usually the test equipment controls the stress to be zero), no significant residual plastic strains can be observed at the end of the load cycle. The application described here was used to study the influence of the TRIP effect. The transformation induced plasticity, TRIP, is a significant plastic strain contribution which can be observed when a mechanical stress acts during the reverse microstructural change from the austenitic to the ferritic phase. When a mechanical stress is applied, a plastic strain occurs which resembles a parallel shift of the graph, see Fig. 5 (right). This effect is neglected in many cases, because the material data are not available, additional experimental runs are required and its influence on the simulation results is still in discussion.

7 MECHANICAL SIMULATION

The results of the thermal analysis were used for the subsequent mechanical analysis. For this purpose a special postprocessing was run that extracts the significant information of each element of the roll model. This information consists of three values including

- the local maximum temperature during the process,
- the austenitization time (time span in the austenitic regime) and
- the cooling gradient (time between 800°C and 500°C, t8/5).

This value tripel was stored for each element for further processing in the VWS tool.

The mechanical simulation was performed with a reduced mesh representing the axial center of the roll. This reduced mesh consisted of one single line of elements from the inner diameter to the outer diameter. This reduced mesh was used to get ideal results for the center portion of the roll without taking end effects into account. The reduced mesh was constrained in axial direction so that plane sections were enforced, with axial displacements suppressed on the right side and axial displacements coupled on the left side. For the material a viso-plastic stress-strain law was chosen which is included in the VWS tool.

The temperatures of the thermal analysis were applied and the STAAZ method activated to idealize the thermal expansion. When the heat-up and cool-down cycle were passed and uniform room temperature was reached, the residual stress distribution was recorded.

Fig. 6 shows the residual stress distribution along the radial direction. The left side of each curve represents the position at the material surface of the roll (outer diameter). The right end of each curve represents a position 40 mm inside the material in radial direction. Both curves show a compression residual stress at the surface which changes to a tension stress at a moderate depth.

Fig. 6 (left) shows the residual stress distribution neglecting the TRIP effect. Fig. 6 (right) displays the residual stress distribution including the TRIP effect.

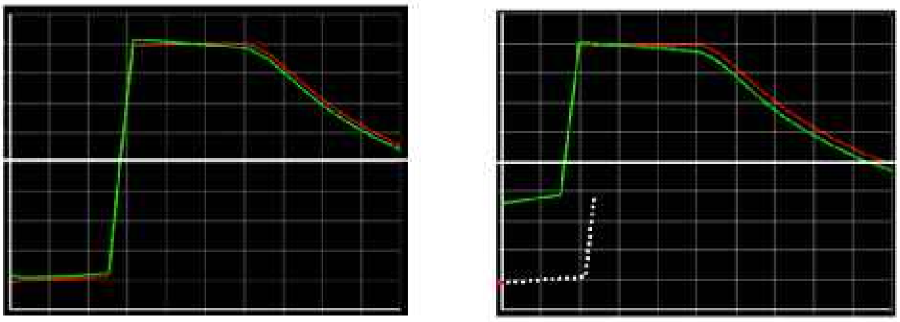

Fig. 6: Residual stress profile neglecting (left side) and including (right side) the TRIP effect

After these initial results were available, the simulation was repeated with slightly adjusted material and process data.

8 RESIDUAL HARDENING STRESS MEASUREMENTS

The residual stress measurements were made by the following methods:

a) X-Ray stress Analyzer (=> surface 0.05 mm deep results)
b) Bore hole method (=> surface 0.1-2 mm deep results)
c) Ring-Core-Method (=> surface 2 – 25 mm deep results)
d) 2-D Measurement from roll end
e) SACHS Method (=> roll center bore to 50 mm below the surface)

We compared the results of all 5 methods and found best matching results in our case with the Ring-Core method. This method gives good data for the surface and the most reliable data for the section of interest 0 to 25 mm depth into the surface. The 2-D end plate test also yielded good results but it always requires an aditional numerical calculation.

9 COMPARISON OF SIMULATION AND MEASUREMENT RESULTS

The following diagram, Fig. 7, shows the simulated stress distrubution (including TRIP) and the measured values measured with the Ring-Core method. The comparison is satisfactory.

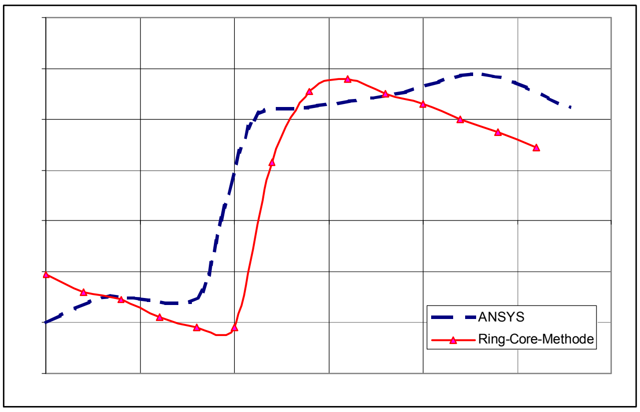

Fig. 7: Resulting residual stress distribution, simulation vrs measurements

This method now allows to optimize the hardening parameters to get the desired optimised residual stress distribution which is needed for the special operating conditions of a roll.

REFERENCES

[1] R. Ossenbrink and V. Michailov, "Simulation der Gefügeumwandlung mit der STAAZ-Methode", 23. *CADFEM Users Meeting*, Bonn, 2.6.7, (2005).